An In-house Manual for Building APEX Projects Using ArcAPEX

S. Teet, A.N. Nelson, J. A. Verser, D.N. Moriasi, J.L. Steiner, P.J. Starks, and P.H. Gowda

A Document of the USDA ARS Grazinglands Research Laboratory

El Reno, OK

Contents

SECTION 1 Introduction

The Agricultural Policy/Environmental Extender (APEX) model is a daily time step model for simulating whole farms or small watersheds to evaluate agricultural production and environmental impacts.

SECTION 1.1 Purpose

The purpose of this document is to provide a step-by-step procedure for creating an ArcAPEX project, and includes guidelines and tips to aid users. This document is a work in progress and was developed primarily for USDA-ARS Grazinglands Research Laboratory (GRL) in-house use and for use by GRL collaborators. This work was partially funded by USDA Office of Environmental Markets.

USDA ARS GRL and the authors are not responsible for use outside the scope described herein. If you have any comments or suggestions for the improvement of this document, please contact Dr. Daniel Moriasi at daniel.moriasi@ars.usda.gov.

SECTION 1.2 User Support

This document was created with reference to ArcAPEX version 0806.10_2.2 and ArcMap 10.2.2. ArcAPEX and ArcMap software are subject to change over time as new versions are released, and use of other versions not referenced here may lead to discrepancies when following the steps outlined in this document.

Note: When creating an APEX project, make sure that your versions of ArcAPEX and ArcMap software are compatible with each other. See the following website for more information:

- APEX user website
  http://apex.tamu.edu/software/
- APEX user support
  http://apex.tamu.edu/support/
- EPIC / APEX Modeling Question and Discussion Google forum
  https://groups.google.com/forum/#!forum/agriliferesearchmodeling

SECTION 2 Preparing ArcAPEX input

Before starting a project, you need to collect as much relevant data as possible. Input data include a digital elevation model (DEM), land use, land management, soils, and weather (especially precipitation and min and max temperature) data. Calibration and validation data may include crop yields, soil water content, components of watershed balance components (e.g., discharge, ET, surface runoff, tile drainage, streamflow/water yield), water quality (e.g., sediment, nitrogen, and phosphorus loads including their species or sources), or other parameters relevant to the objectives of the simulation project being built. The following list shows the types of data either required or desirable to implement an APEX run. Utilize user-supplied data, if available. **For a complete list and format of input and output data, refer to the apex0806-user-guide, available at http://apex.tamu.edu/documentation.**

1. Land use
2. Soils map and characterization
3. Land Management
   3.1. Irrigation information
   3.2. Fertilization information
   3.3. Pesticides information
   3.4. Tillage practices
   3.5. Planting/harvesting dates
   3.6. Grazing / livestock data
4. Digital Elevation Map (DEM)
5. Daily weather station data
   5.1. Wind speed
   5.2. Wind direction
   5.3. Temperature
   5.4. Solar radiation
   5.5. Precipitation
   5.6. Relative humidity
6. Observed data that may be used for calibration and validation
   6.1. Soil moisture
   6.2. Streamflow/water yield
   6.3. Drainage flow
   6.4. Surface runoff
   6.5. Nutrient data
   6.6. Sediment data
   6.7. Evapotranspiration (ET)
   6.8. Crop yields / biomass
   6.9. Point sources

SECTION 2.1 Required spatial datasets for building an ArcAPEX project

There are three spatial data sets, referred to as layers, required to set up an APEX project -- a DEM, land use, and soils. These layers should be ArcGIS shape files in order to make the project setup easier.

Before starting a project, all required spatial data layers must be in the same projected coordinate system. If using the built-in STATSGO database, the DEM and land use layers will need to be in Albers Equal Area projection to match the projection of the STATSGO data set. This is easily accomplished by opening a blank ArcMap project and adding the STATSGO **statsgo_grd** shape file, located in your ArcAPEX directory (**C:\APEX\ArcAPEX\Databases\ SWAT_US_Soils.mdb\statsgo_grd)** if you used the

default directory when installing ArcAPEX. The data frame properties will default to the **statsgo_grd** feature since it is the first layer added. Next add the DEM and land use layers and project them to Albers Equal Area as needed.

SECTION 2.1.1 DEM

The DEM is used in the watershed delineation process. The DEM layer MUST BE a raster shapefile. As APEX is intended for smaller, farm/field level watersheds, higher resolution DEMs are ideal. One main source for DEM data is the USGS National Elevation Dataset (NED), using 'The National Map Viewer'.

Tip: If you need to combine multiple DEMs, use **ArcToolbox > Data Management Tools > Raster > Mosaic Dataset** to create a new File Geodatabase (within "**Create Mosaic Dataset > Output Location**," in the file folder icon). Give the Mosaic Dataset an appropriate name, choose a coordinate system, select 32-bit signed minimum (to avoid null values), then press **OK**.

Then select **Add Rasters to Mosaic Dataset**. In '**Input Data**', select "**dataset**" then navigate and select the rasters with which you want to create the Mosaic Dataset. In the advanced options section, tick '**Build Raster Pyramids**', '**Calculate Statistics**,' and '**Build Thumbnails**'. Click **OK** to add the rasters to the mosaic dataset.

Note: Your combined raster may not show the DEM color gradations. This is ok, the data are still there. You may adjust the display options, but color gradations are not necessary to build the project.

SECTION 2.1.2 Land Use Layer

This layer can, at the start, be either raster or vector (polygonal) layer; if a vector layer is used, it should be converted to a raster layer during the subarea analysis in ArcAPEX. Land use complexity may be as simple as delineation between agriculture, urban and / or other land uses or as complex as a delineation of several fields having different crops and management practices.

SECTION 2.1.3 Soils Layer

Unless you are planning on using the built-in STATSGO database, you will need to have a georeferenced soils coverage. This coverage can, at the start, be either a raster or vector (polygonal) layer; if a vector layer is used, it should be converted to a raster layer during the subarea analysis in ArcAPEX

The soils coverage must also have a lookup table and corresponding database with detailed information for each soil included. The best format for the lookup table is a comma delimited text file that includes a "Value" field and a field for the type of identification used (Table 2.1), where "Value" is an ID number. See Appendix B for more details.

Table 2.1 Example of soils lookup table format where "value" is an id number and "S5ID" is the soil identification code used (in this case, a state id plus a four-digit numeric code)

"Value","S5ID"
0,OH0262
1,OH0262
2,OH0047
. . .

SECTION 3 Getting started with ArcAPEX

To start the ArcAPEX Interface:

1. Start **ArcMap** and open an empty document
2. If this is your first time using ArcAPEX: From the Customize menu, click **Toolbars** and select the **ArcAPEX Toolbar**. The ArcAPEX Toolbar will appear in your ArcMap window (Figure 3.1).

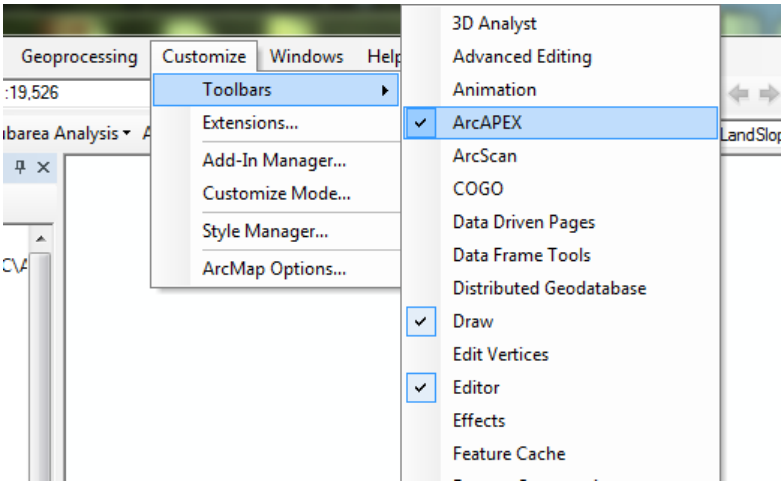

Figure 3.1 Turning on ArcAPEX toolbar in Toolbar customization.

SECTION 3.1 Previously Established ArcAPEX Projects

If you have files from a previously established ArcAPEX project, you may use the **Open APEX Map Document** command to open the map document associated with the project, which will also open the APEX project. The remainder of this section will focus on the creation of a new project.

SECTION 3.2 New ArcAPEX project

To create a new ArcAPEX project:

1. From the **APEX Project Setup** menu, click the **New APEX Project** command (Figure 3.2).

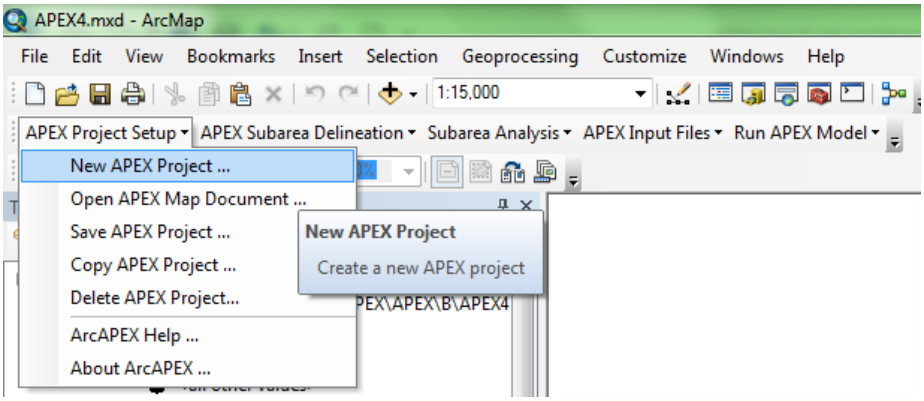

Figure 3.2 ArcAPEX project setup

2. A dialog box will appear and ask if you want to save the current document.

3. After choosing an appropriate response, the **APEX Project Setup** dialog box will appear (Figure 3.3). The dialog box will contain initial default values for an **APEX Project Directory**, **APEX Project Geodatabase**, **APEX Raster Storage**, and **APEX Parameters Geodatabase**.

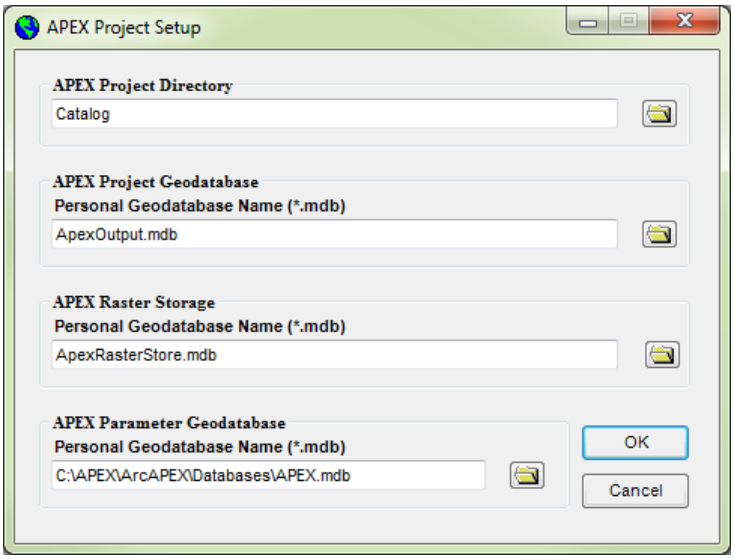

Figure 3.3 ArcAPEX project directory and database setup box

4. Choose an **APEX Project Directory** by clicking on the file browse button to the right of the text box. The Project Directory will be the location where all your APEX project files will be stored. You can also create a new folder within this function. Be sure to select or create a new, empty

folder in which to build your project. Once you have done this, the **APEX Project Setup** will automatically fill in the additional boxes.

5. Change the name for the **APEX Project Geodatabase** (**Optional**). By default, the interface will set the name of the geodatabase as the same name as the project folder.

6. Change the name of the **APEX Raster Storage Geodatabase** (**Optional**). By default, the interface will set the name of the geodatabase as the same name as the project folder.

7. **Change the name of the APEX** Parameter Geodatabase (**Optional**). By default, the **APEX.mdb** geodatabase in your ArcAPEX installation folder will be chosen. Some users may wish to maintain multiple version of this database, in which case they would be able to select an alternative database name here.

8. Click **OK** to continue.

Now add the DEM, land use, and soils layers that you previously created/edited. These will be used in the APEX project creation.

SECTION 4 ArcAPEX Project Setup

SECTION 4.1 APEX Subarea Delineation

Next, click the **APEX Subarea Delineation** tab in the **APEX toolbar** (Figure 4.1) and select **Automatic Subarea Delineation > APEX Standalone Delineation**.

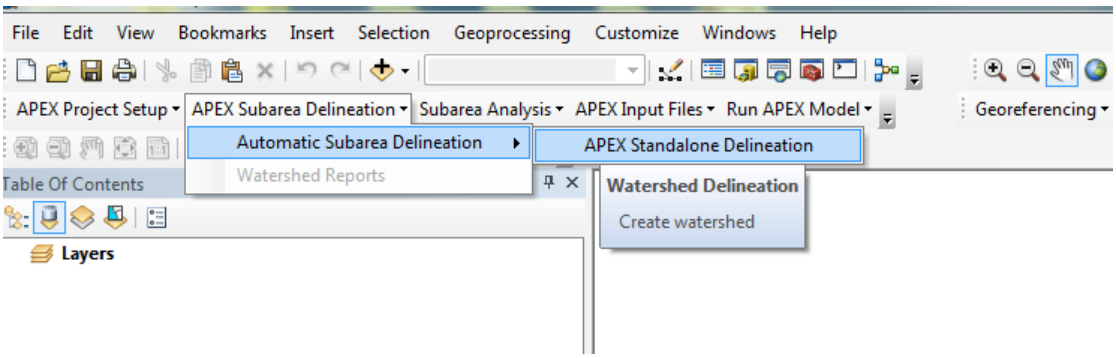

Figure 4.1 APEX subarea delineation tab

This will bring up the **Subarea Delineation** dialog box (Figure 4.2).

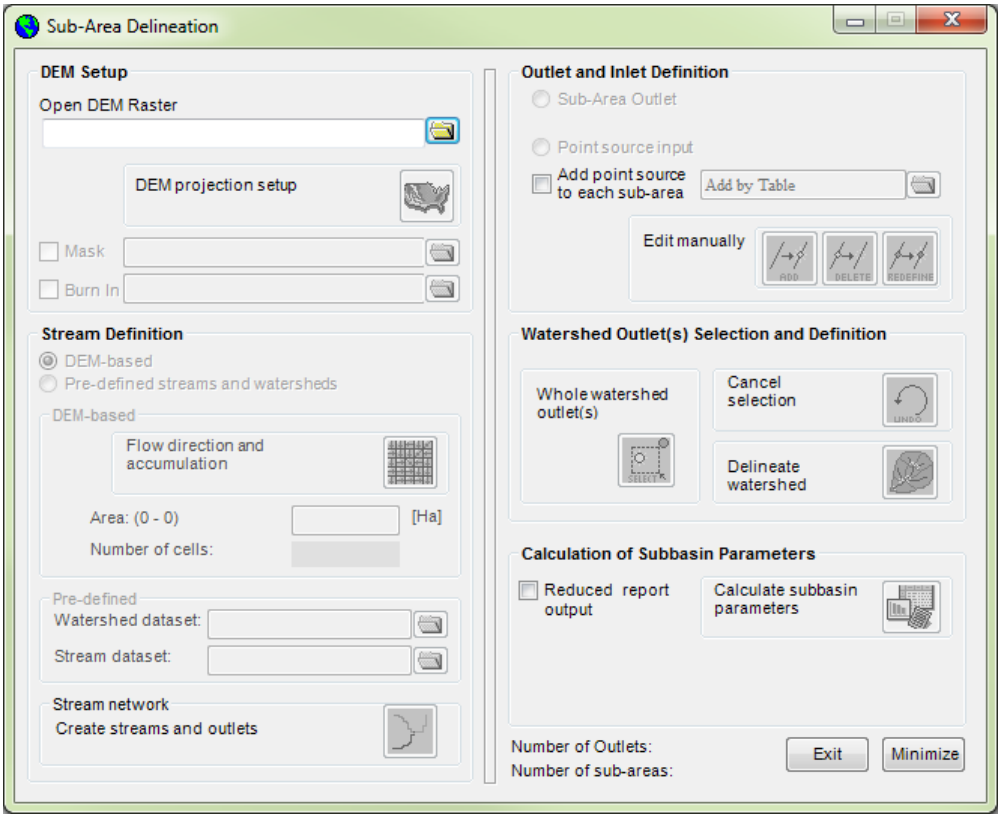

Figure 4.2 Subarea Delineation box

Click on the folder under **Open DEM Raster** (Figure 4.2), select **Load from Disk** and click **OK** (Figure 4.3). Navigate to your DEM (disk) or select it from your table of contents (map) and click **add**.

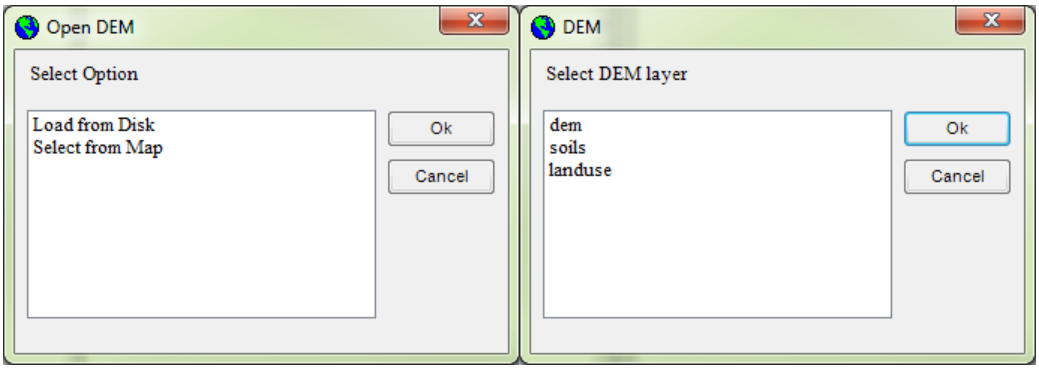

Figure 4.3 Open DEM selection box and DEM layer selection box

Next, under **Stream Definition** in the **Sub-Area Delineation** box (Figure 4.4), make sure **DEM-based** is selected and in the **DEM-based** section, click the grid for **Flow direction and accumulation**. This will automatically fill in the **Area** and **Number of cells** boxes.

Note: It is possible to edit the Area Box; the larger the number of inputs the fewer subareas that will be created.

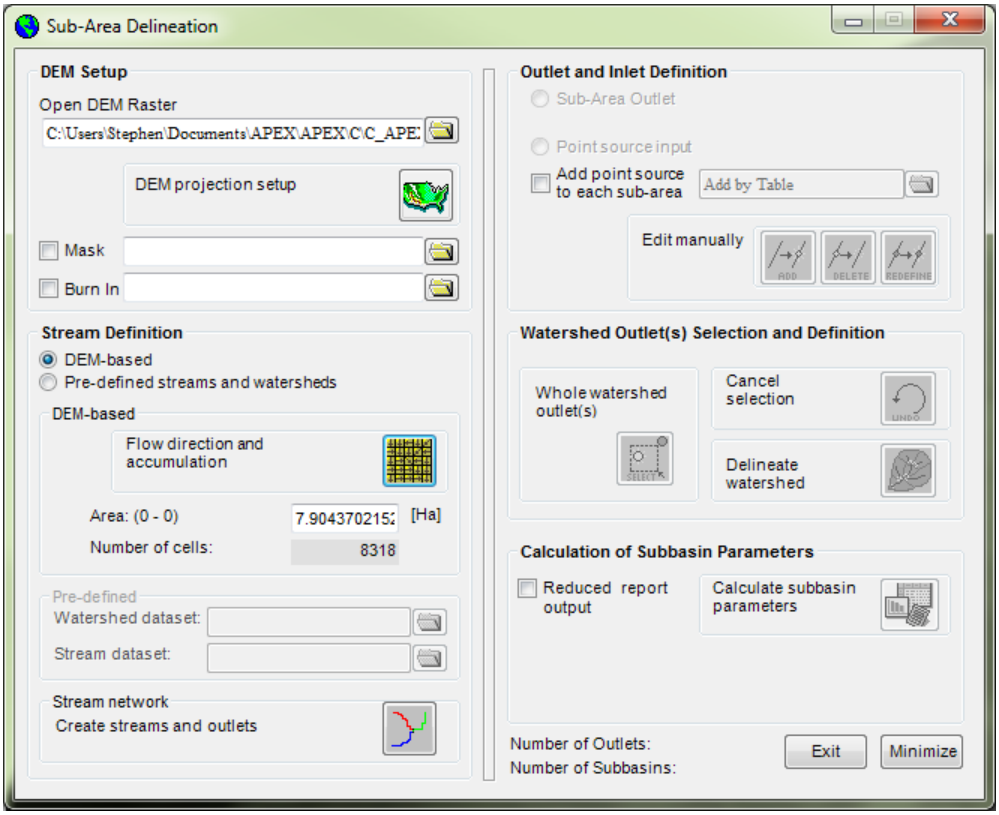

Figure 4.4 Sub-Area Delineation box with Stream Definition and Stream Network options

In the **Stream network** section, the **Create streams and outlets** button is now highlighted, click it to run the stream delineation.

Now the rest of the sections in the DEM Setup box are available for use. For a typical model build, it will be necessary to run through the delineation first, then, if there are too few or too many (most likely) sub-areas, go back through the delineation and use the **Outlet and Inlet Definition** section to add, delete or edit points.

If there are outlets disconnected from the rest of the watershed, such as the ones on the north and southeast boundaries of Figure 4.5, take this opportunity to delete them.

In the **Watershed Outlet(s) Selection and Definition** section, click the **Whole watershed outlet(s)** button. A window will pop up and tell you to use the left mouse button to select your outlet, click **OK**. The **Sub-Area Delineation** box will now go to the background while you select the outlet(s) for your watershed, click and drag to create a box around the node or nodes (Figure 4.5).

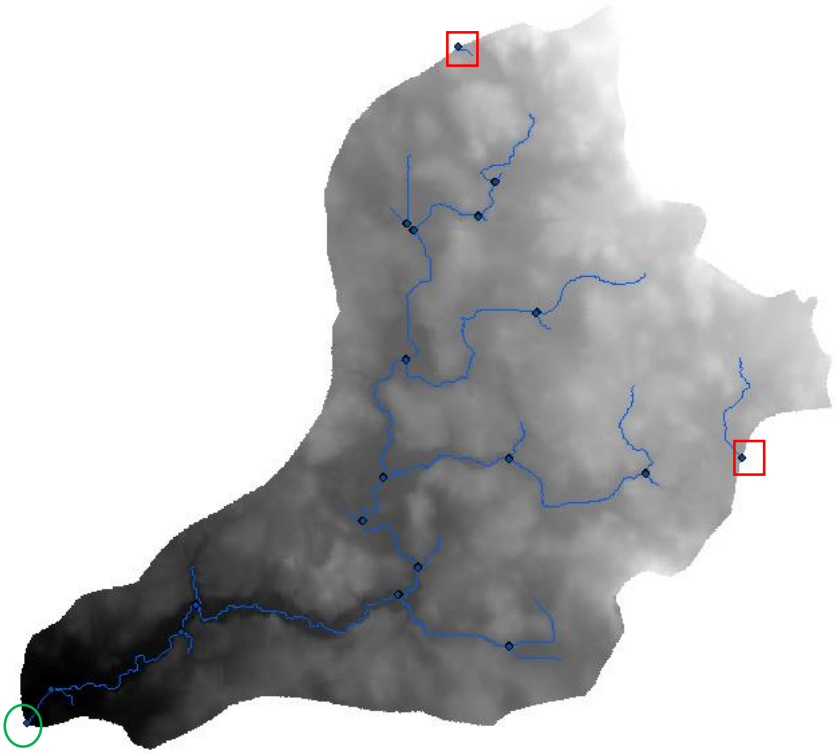

Figure 4.5 Example of outlying outlets to be removed (red squares) and subarea outlet selection (green circle)

The **Cancel selection** and **Delineate watershed** boxes are now available. Click the **Delineate watershed** button.

The delineation is now complete (Figure 4.6). Click the **calculate sub-basin parameters** button and then exit the **Sub-Area Delineation** box.

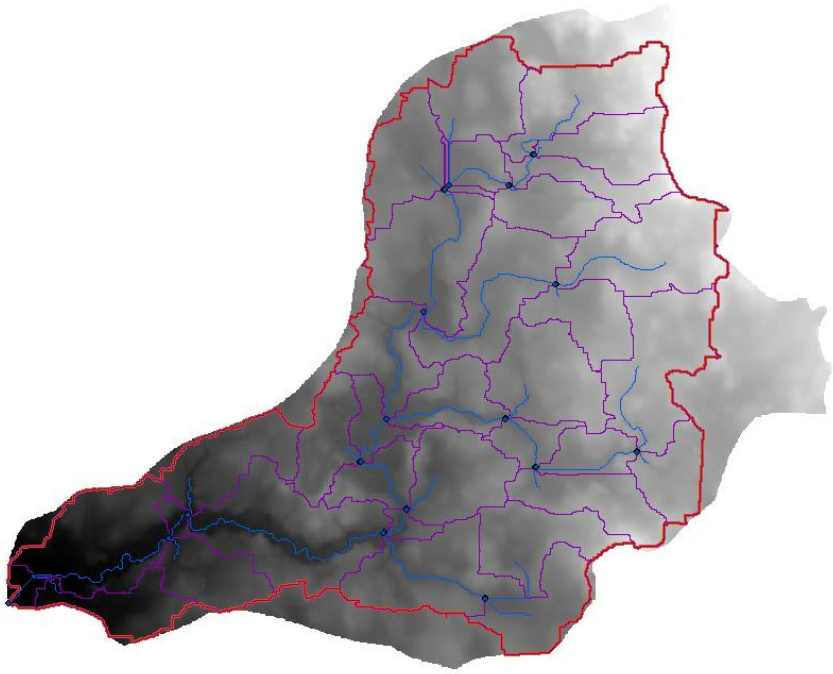

Figure 4.6 Example of delineated watershed

SECTION 4.2 Subarea Analysis

SECTION 4.2.1 Land Use/Soils/Slope Definition

Note: The following section is often the most frustrating and there may be need for troubleshooting. If at any time there is an error halting progress, one or more of the following steps may be required:

- Close and restart the subarea analysis step, or
- Close and restart your APEX project and/or ArcMap, or
- Close and restart your computer, or
- Start your project over from the beginning

The next step in the APEX project setup using ArcAPEX is Subarea Analysis. First, load the land use and soils layers into your map, if you haven't already done so. Under the **Subarea Analysis** dropdown click **Land Use/Soils/Slope Definition**.

In the **Land Use Grid** section of the Land Use Data tab, click the folder, select **load from map**, then choose your land use map from the list and click **OK** (Figure 4.7).

In the **Choose Grid Field** section, click the drop down, choose **value** and click **OK**.

Now, either add a land use Look-Up Table (Appendix B) or manually classify the different land use areas.

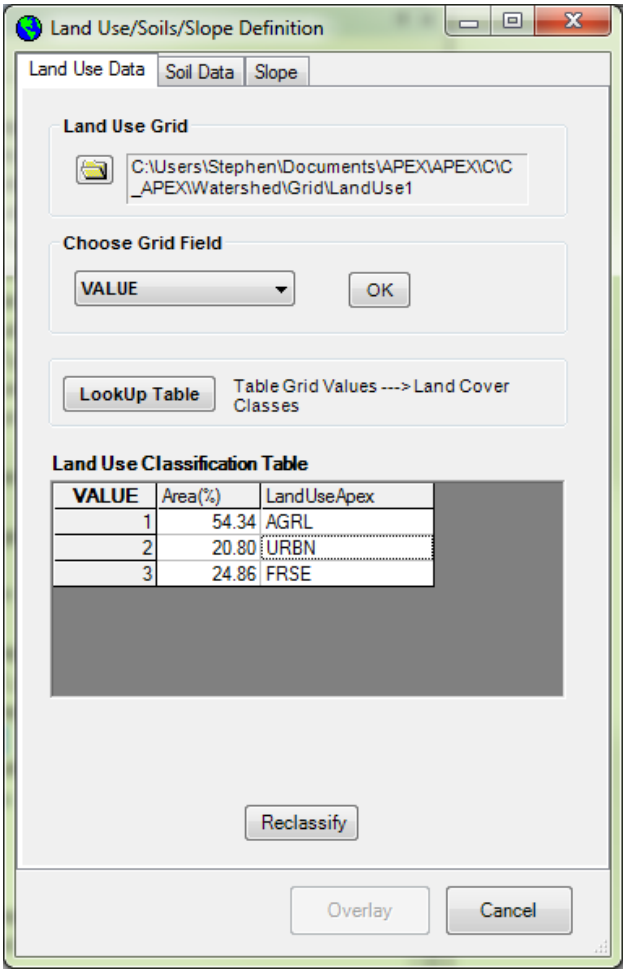

Figure 4.7 Land Use Data tab of the Land Use/Soils/Slope Definition box

To manually classify the land uses, double click the empty **Land Use APEX** boxes one at a time. This will bring up a new box titled **APEX Land Use** (Figure 4.8); click the **Land Cover Database** drop down and choose crop. Click **OK**.

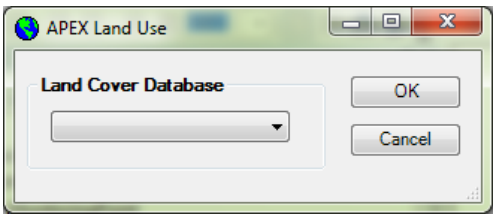

Figure 4.8 APEX Land Use box

A new box titled **APEX-Land Cover/Plant** will pop up (Figure 4.9). Choose the appropriate crop from the list for the selected land use area and click **OK**. If the selected area represents multiple agricultural uses, choose Agricultural-General→AGRL, otherwise pick the crop if available. Urban land cover is under Pavement→URBN.

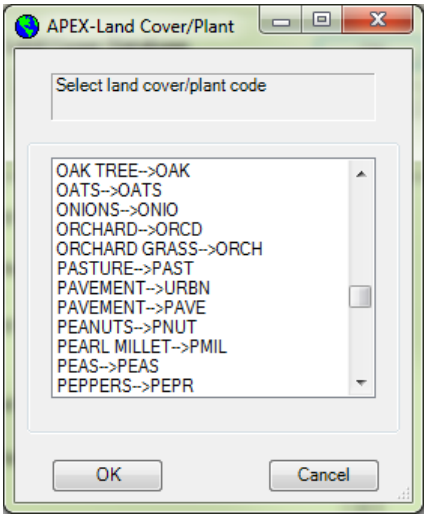

Figure 4.9 APEX-Land Cover/Plant box

Tip: It is recommended you create a lookup table in case you need to restart at any point during the Land Use/Soils/Slope Definition step (Appendix B).

Once you have classified all of your land use areas, click **reclassify**.

Now move to the **Soil Data** tab in the **Land Use/Soils/Slope Definition** box. Click the folder in the **Soils Grid** section. If no soils data were supplied for the project, select **Load ArcAPEX US STATSGO** from disk; if there was, select **Load Soils dataset(s)** from the map in the **Select Soils Data** box and click **Open** (Figure 4.10).

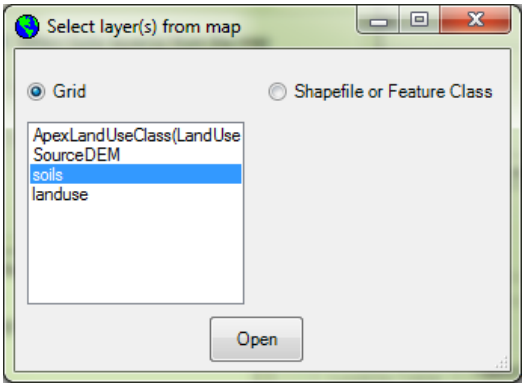

Figure 4.10 Select layer(s) from map box

In the **Select layer(s) from map** box, click **Grid**, then select the soils map and click **OPEN**, the subsequent pop-up window will say that the layer was successfully loaded and clipped if there were no problems. Click **OK**.

This brings up a box titled **Land Use/Soils Grid Code**. From the drop down menu, select the field from your map database that defines the soils (Figure 4.11). Click **OK**.

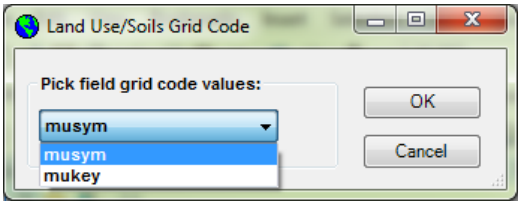

Figure 4.11 Land Use/Soils Grid Code box

In the **Options** section of the **Soil Data** tab, choose the **soil identification type** to be used. Now, either add a lookup table (Appendix B) or fill in the values in the empty column(s) (Figure 4.12).

Click **Reclassify**.

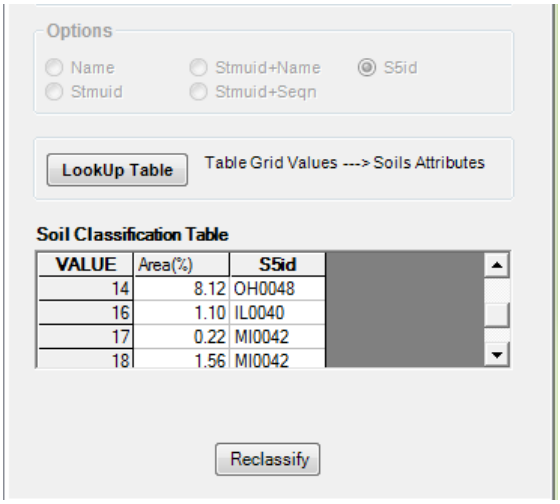

Figure 4.12 Example of soil classifications using the S5id identification

Click the **Slope** tab in the Land Use/Soil Data/Slope box (Figure 4.13). In the **Slope Discretization** section, pick either **Single Slope** or **Multiple Slope**. If Multiple Slope is selected, complete the **Slope Classes** section (you can have 2 – 5 slopes). Click **Reclassify**.

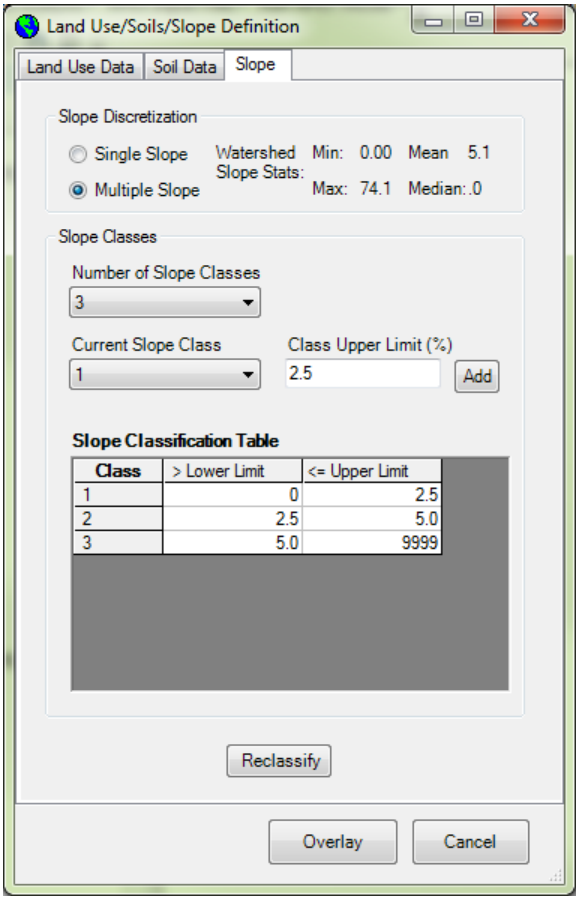

Figure 4.13 Slope definition

The **Land Use/Soils Data/Slope** section is now complete and the **Overlay Button** is activated. Click it to finalize this process.

SECTION 4.2.2 Subarea Definition

Under the **Subarea Analysis Dropdown**, pick **Subarea Definition**. In the **APEX Subarea Definition** box (Figure 4.14) pick a **Subarea Definition Option** (either Dominant Land Use, Dominant Soil, Dominant Slope or Dominant Land Use/Soil/Slope Combination) from the list and click **OK**. This will complete the subarea analysis and allow access to Subarea Analyses Reports if needed.

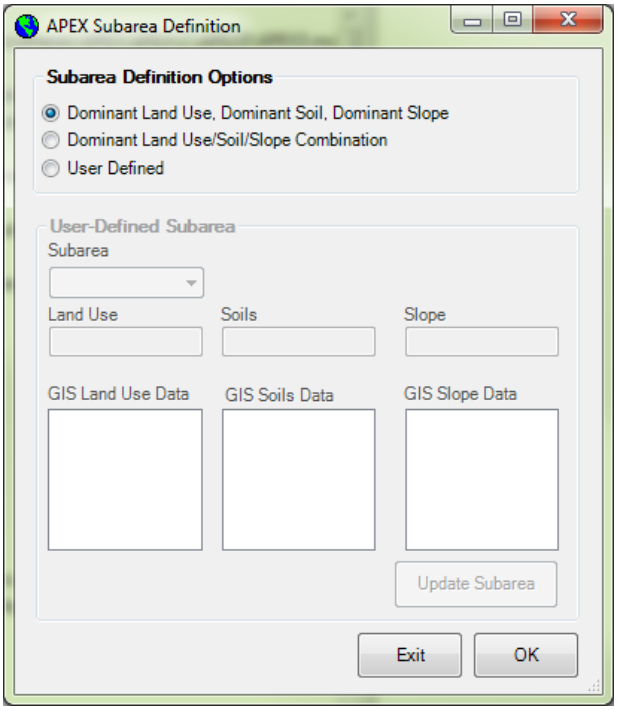

Figure 4.14 APEX Subarea Definition box

SECTION 5 APEX Input Files

Under the **APEX Input Files** dropdown on the toolbar, pick **Edit APEX Databases**. This brings up a box with a list of databases in APEX that can be edited or built by the user (Figure 5.1).

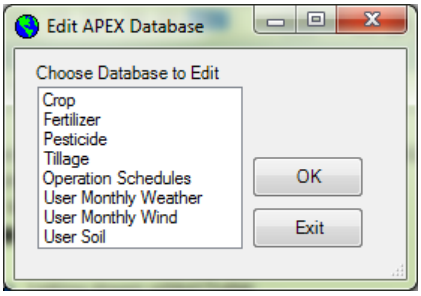

Figure 5.1 Edit APEX Databases box

SECTION 5.1 Add/Edit Fertilizer

The list of fertilizers already in the APEX database is limited and may not include all needed fertilizers. To add a fertilizer to the database, highlight **Fertilizer** in the **Edit APEX Database** window (Figure 5.1) and click **OK**.

The **Edit Fertilizer Database** window will pop up (Figure 5.2). This includes the complete list of available fertilizers in the scroll window on the left side of the window. To add a new fertilizer, select a fertilizer and click the **Add** button below the fertilizer list.

Note: Many of the fertilizer list entries have the format xx-xx-xx (such as 14-46-00) representing the percent nitrogen (N), phosphorus (P) and potassium (K) in the fertilizer mixture.

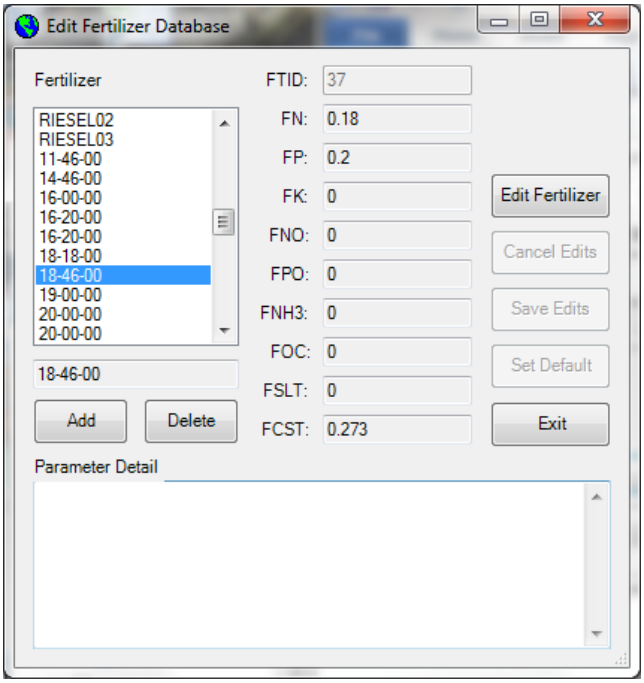

Figure 5.2 Edit Fertilizer Database window

The **Fertilizer Name** window will pop up (Figure 5.3), enter the name of the fertilizer to be added here. Click **OK**.

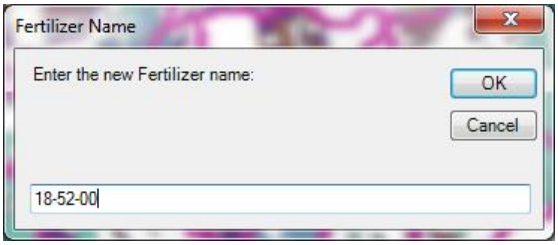

Figure 5.3 Fertilizer Name window

The fertilizer parameters in the **Edit Fertilizer Database** are now available to be edited (Figure 5.4). Enter the values for the nutrient fractions of the new fertilizer, as well as the cost, and click **Save Edits**.

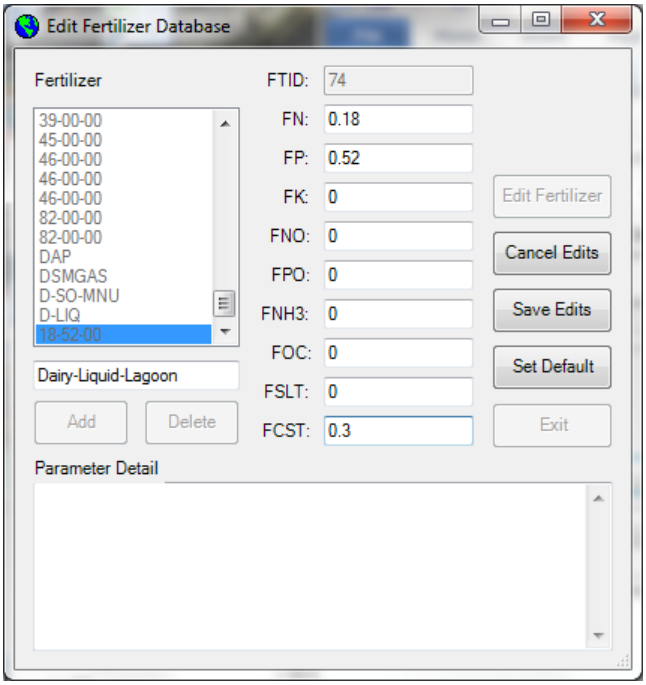

Figure 5.4 Active Edit Fertilizer Database window

Once all new fertilizers have been added, click Exit.

SECTION 5.2 Add/Edit Operation Schedules

ArcAPEX comes with a limited set of usable operation schedules. This makes it necessary to create operation schedules for all crops and rotations being used in the project.

To build a new operation schedule, choose **Operation Schedules** in the **Edit APEX Database** window (Figure 5.1) and click **OK**. The **Edit Operation Schedules** window will open with list of all the available operation schedules on the left side (Figure 5.5).

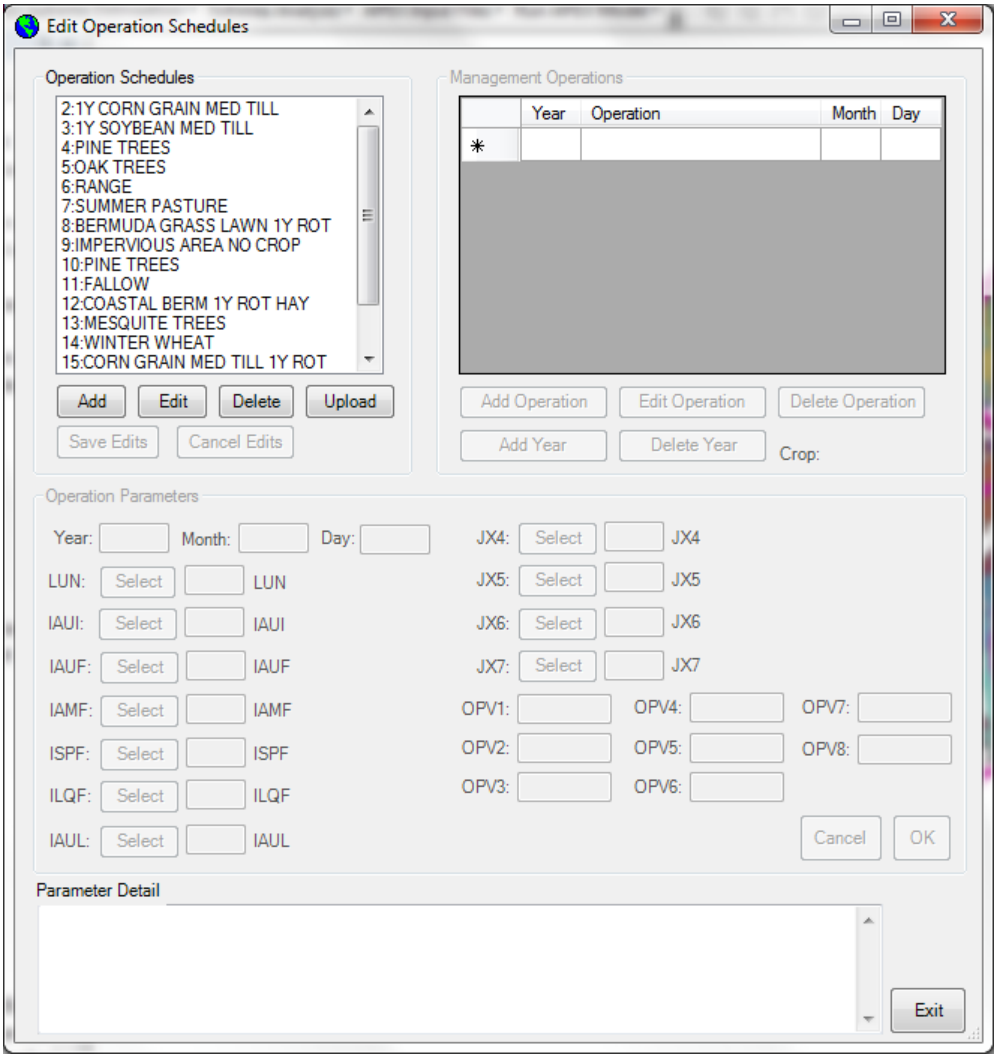

Figure 5.5 Edit Operation Schedules window

To add a new operation schedule, click **Add** under the list. The **Operation Schedule Name** window will open (Figure 5.6); enter the name to be used by APEX in the text file (limited to eight characters) and click **OK**.

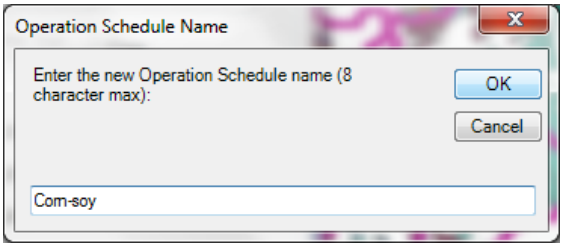

Figure 5.6 Operation Schedule Name window.

The **Operation Schedule Description** window will pop up next (Figure 5.7). In this window, enter a more descriptive name for the new operation schedule; this is the name that will be seen in the Operation Schedules list (Figure 5.9). Click **OK**.

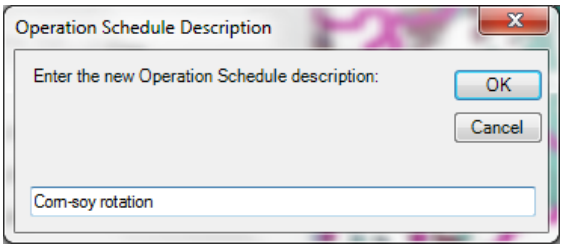

Figure 5.7 Operation Schedule Description window

Operations will now need to be added to the new operation schedule. Select the new operation from the **Operation Schedules** list and click the **Edit** button beneath the list (Figure 6.5). To add a new operation, click **Add Operation** under the **Management Operations** box. The **Choose Operation** window will pop up (Figure 5.8). Choose the operation type from the **New Operation** list on the left and the specific operation from the **Operation list** on the right. Click **OK**.

The first example is for the addition of a fertilize operation. Figure 5.8 shows the addition of a liquid fertilizer application operation.

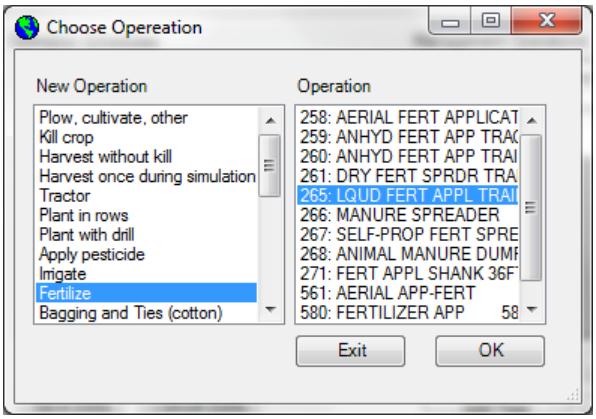

Figure 5.8 Choose Operation window

In the **Edit Operation Schedules** window, the **Operation Parameters** at the bottom are now available (Figure 5.9).

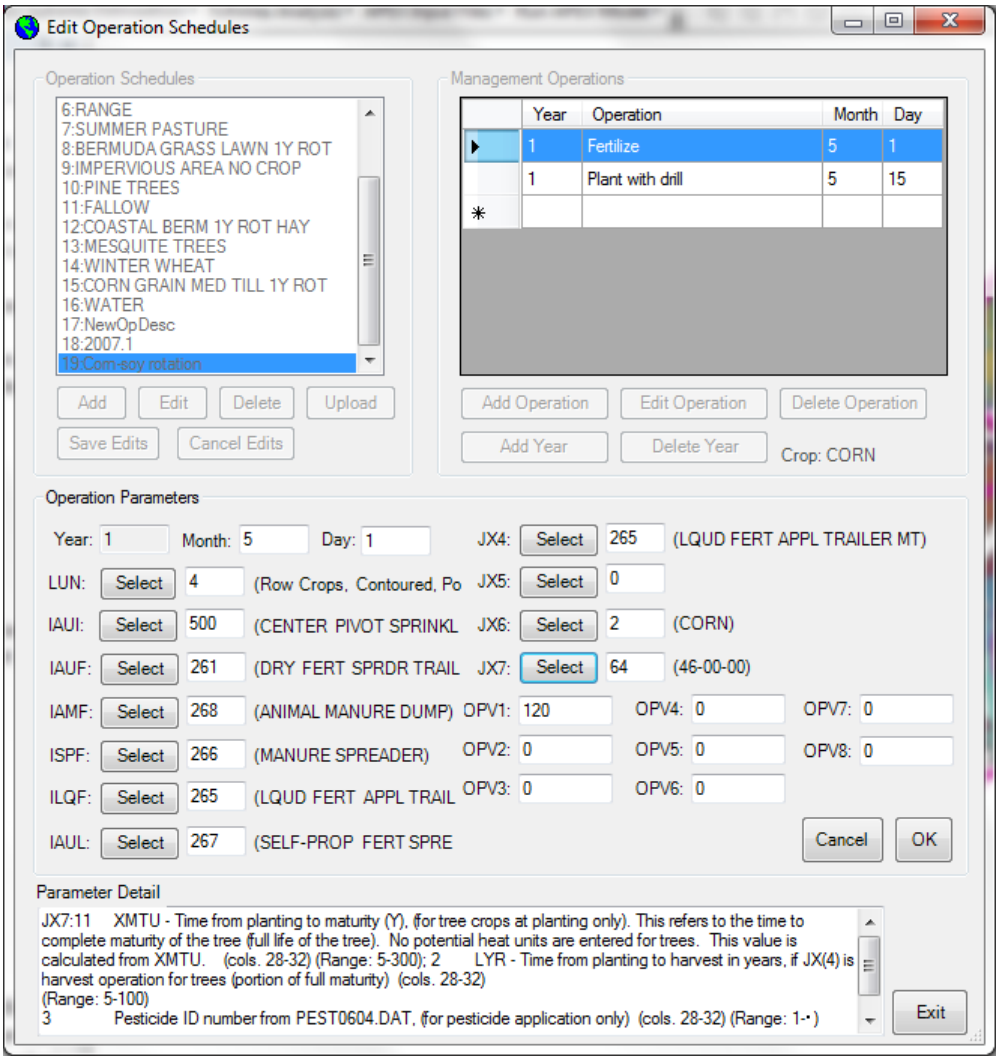

Figure 5.9 Active Edit Operation Schedules window for adding/editing a new fertilize operation

Based on the selections made in the **Choose Operation** window, many of the parameters are filled with set and/or default values.

The JX and OPV boxes, shown in Figure 5.9, represent different things for different operations. For fertilizer, JX4 is the identification number of the type of application for the tillage file (*.TILL).

Note: For fertilizer operation, OPV1 represents the amount of fertilizer being used in kg/Ha. This does not have a set default value and must be given for the APEX model to run. For all operations, there needs to be a value for the equipment used in box JX5, this is often a tractor type.

TIP: To see a description of a specific parameter, double click the number box for that parameter. The description will show up in the Parameter Detail box at the bottom of the Edit Operation Schedules window.

The next example represents adding a planting operation. Just like for the fertilizer, click the **Add Operation** button, choose “Plant in rows” or “Plant with drill” in the **Choose Operation** window under **New Operation**, then select the planting method under **Operation** (Figure 5.10). Click **OK**.

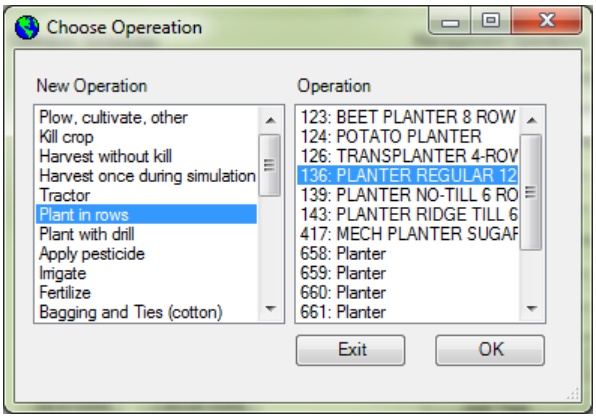

Figure 5.10 Planting crops in the Choose Operation window

As with the fertilize operation, the **Operation Parameters** box is now available. Edit the planting date and any other parameters that are different than the default (Figure 5.11).

Note: When adding crop planting operations, it is important to make sure the right crop has been selected in the JX6 box. It is also important to enter the number of seeds or plants per square meter or per hectare if the amount per meter squared is less than one (OPV5). For planting, OPV1 is the potential heat units. For corn and soy, this value is 1184 and for wheat it is 1200.

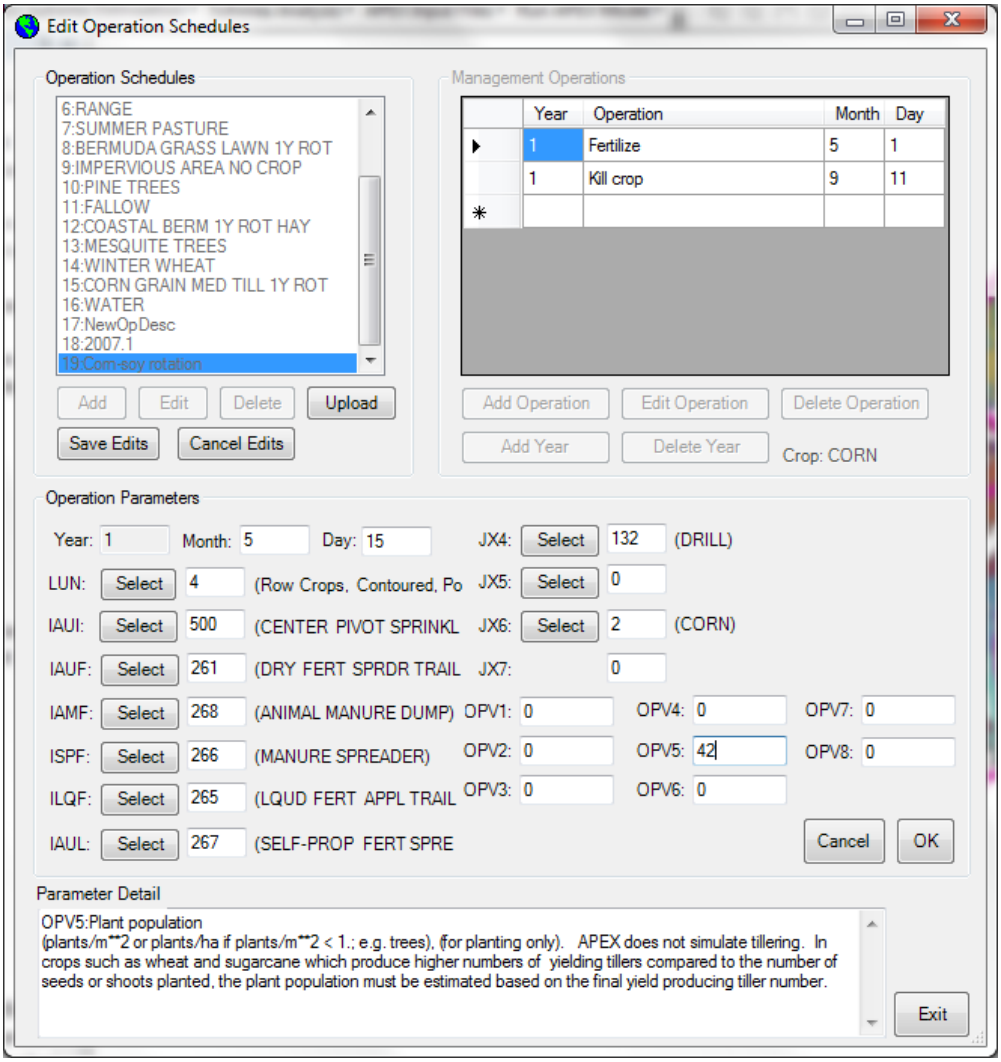

Figure 5.11 Active Edit Operation Schedules window for adding/editing a new plant operation

Once all operations for the crop/rotation have been added, click **Save Edits**. Now, either add a new operation or click **Exit**.

SECTION 5.3 Define Weather Stations

To generate or import weather data, choose **Define Weather Stations** under the **APEX Input Files** dropdown. This brings up the **Define Weather Inputs** window (Figure 5.12).

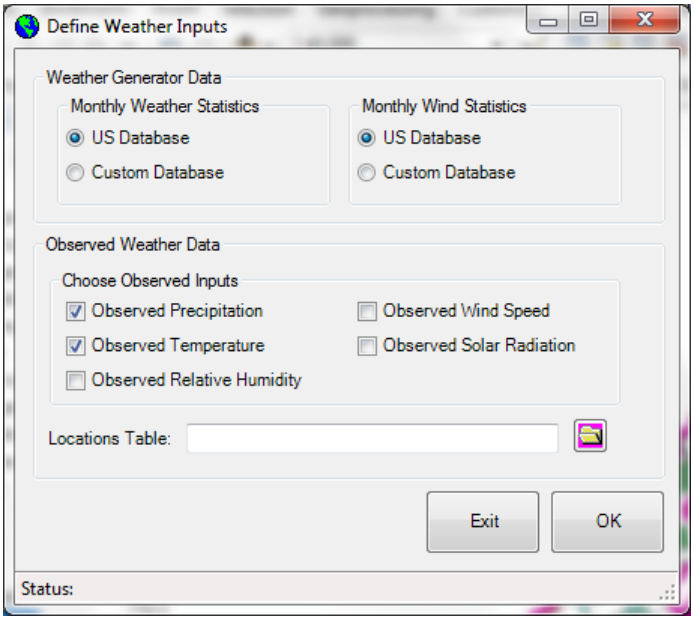

Figure 5.12 Define Weather Inputs window

To use the weather generator, check either **US Database** or **Custom Database** in the **Monthly Weather** and **Wind Statistics** boxes. To use observed data, check the boxes by the available measured data in the **Observed Weather Data** box.

Note: Observed weather data should not be used unless precipitation and temperature values are available.

One or more weather stations may be used. For example, there may be a weather station in each field or just one near the watershed border. Either way, APEX will pick one station per subarea.

A locations table must be used even if there is only one weather station being used for observed weather data. This table should include an ID, file name, latitude, longitude, and elevation (in meters) in a comma delimited format (Table 5.1). Click the folder by the **Locations Table** box to navigate to the locations file. Click **OK**.

Table 5.1 Weather station locations table example.

ID,NAME,LAT,LONG,ELEVATION
1,BucyrsOH.txt,40.816724,-82.966576,291
2,Fremont1.txt,41.33333,-83.11667,182.9
3,GalionWW.txt,40.7231,-82.7999,356.6
4,TiffinOH.txt,41.116667,-83.166667,226

SECTION 5.4 Write Default Inputs

Under the **APEX Input Files** dropdown on the **ArcAPEX toolbar**, select **Write Default Inputs**. In the **Write Default Inputs** window, click the **select all** button and click **OK** (Figure 5.13).

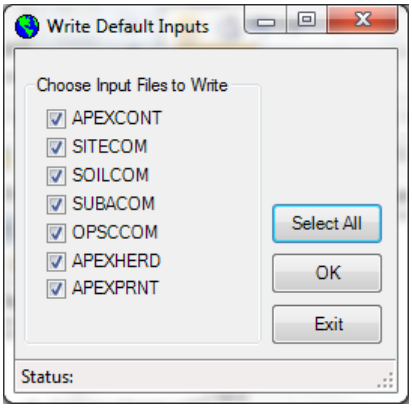

Figure 5.13 Write Default Inputs window

Tip: If an error message occurs during writing default files step, try writing each default file one at a time to narrow down where the problem may be occurring.

SECTION 5.5 Edit Input Files

Next, the contents of the input files need to be reviewed and edited before writing them and making the model run. Choose **Edit APEX Inputs** under the **APEX Input Files** dropdown. This brings up the Edit APEX Inputs window (Figure 5.14) with a list of input files that can be edited.

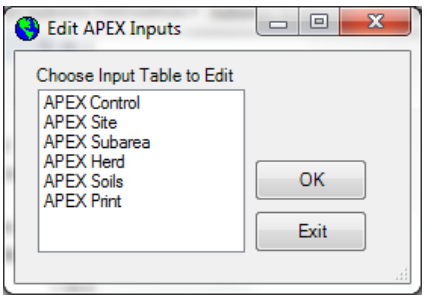

Figure 5.14 Edit APEX Inputs selection window

Choose **APEX Control** and click **OK**. The **Edit Control File** window opens to the **Control Page 1 Parameters** tab (Figure 5.15).

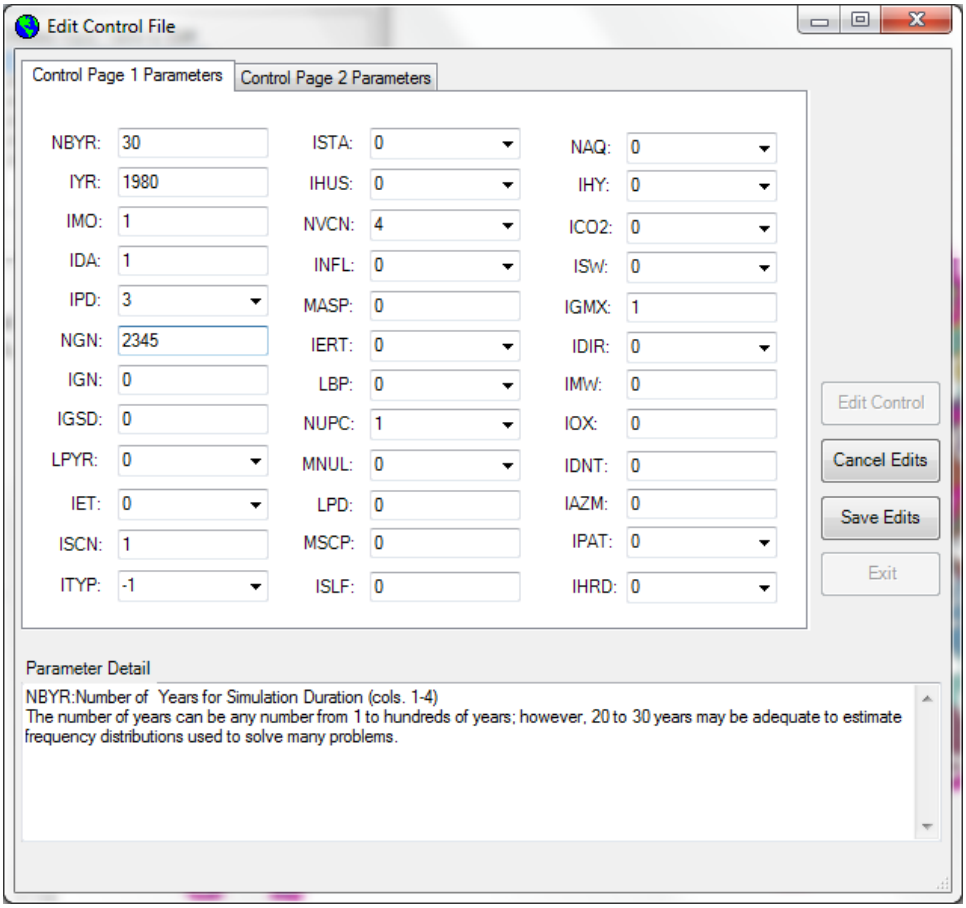

Figure 5.15 Edit Control File window, page 1 parameters

Most edits that need to be made to the control file should be in the **Control Page 1 Parameters** tab. The number of years of simulation (NBYR), initial year of simulation (IYR), initial month (IMO), and initial day (IDA) should be set at this point as the default is universal regardless of data entered to this point. This tab also contains some of the variables used for sensitivity analysis and calibration of the model so any edits should be noted for later reference.

The **Control Page 2 Parameters** tab (Figure 5.16) contains many of the variables used in the sensitivity analysis and calibration of the model and should be left to default values.

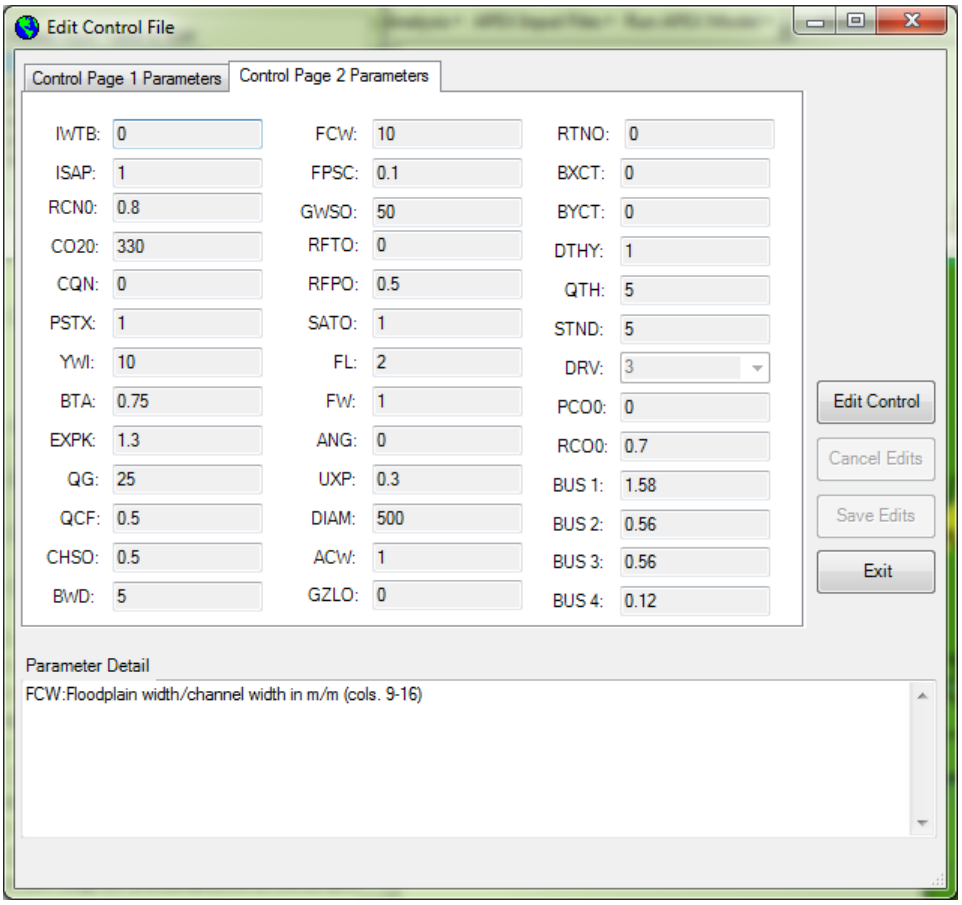

Figure 5.16 Edit Control file window Control Page 2 Parameters tab

The **Edit Site File** window (Figure 5.17) primarily deals with the locations of the outlet and weather stations and should be correct from the model build.

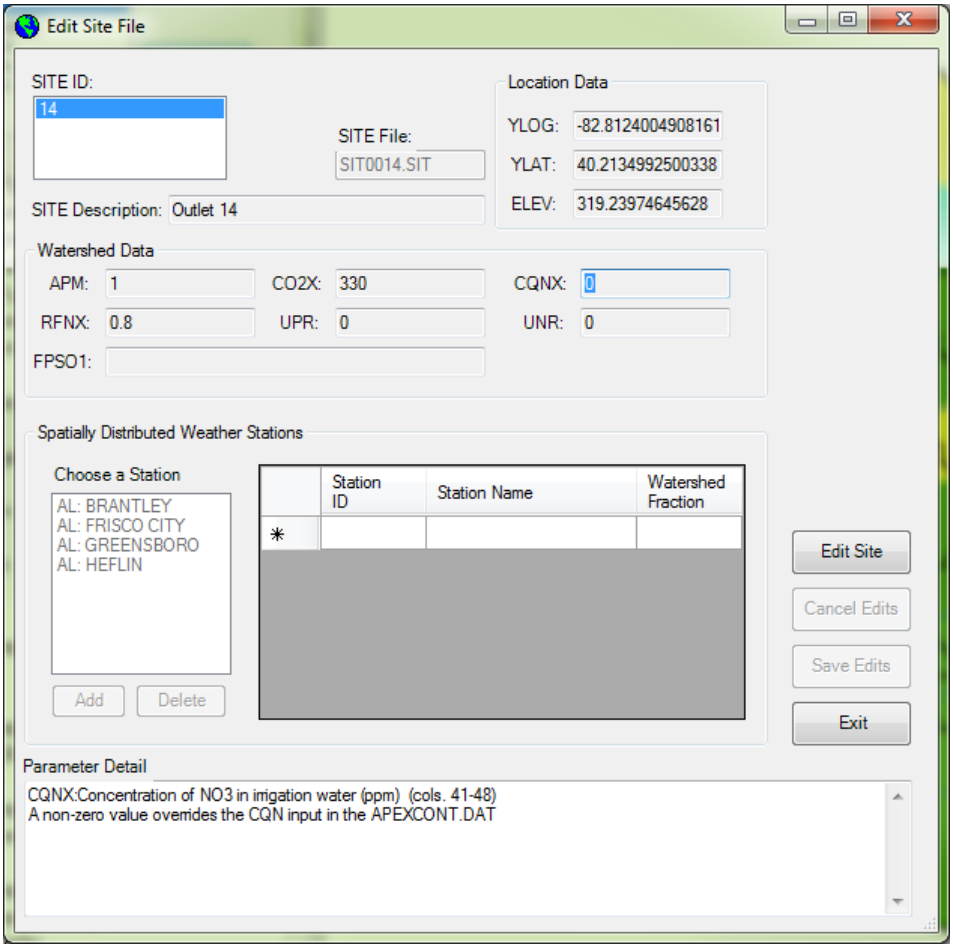

Figure 5.17 Edit Site File window

The **Edit APEX Subarea** window (Figure 5.18) allows several parameters to be edited separately by subarea. If management files were built correctly and applied to the correct subareas, the information in this window should also be correct. Two variables in the top portion of the window, SNO (water content of existing snow) and STDO (standing crop residue), allow for the input of initial conditions of the subarea and can be edited if needed.

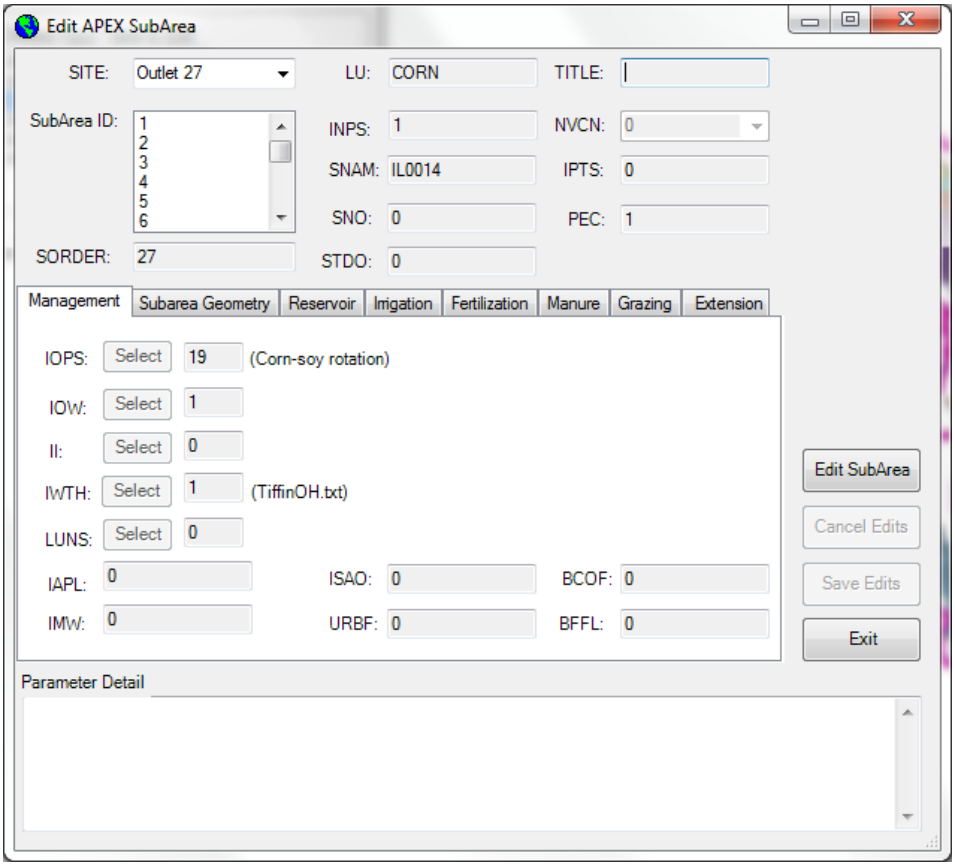

Figure 5.18 Edit APEX SubArea window

The **Edit Print File** window (Figure 5.19) is used to select which output files are printed during the model run. To determine which files may be needed, it is best to refer to the **APEX 0806 User Guide**.

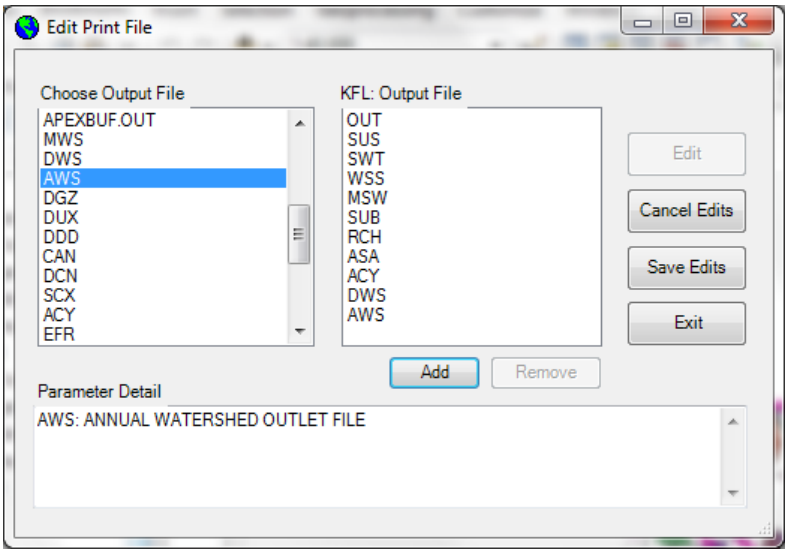

Figure 5.19 Edit Print File window

SECTION 5.6 Write Input Files

Once all needed edits are made to input files, open the **Write APEX Input Files** window under the **APEX Input Files** dropdown (Figure 5.20). Click **Select All** then click **OK**.

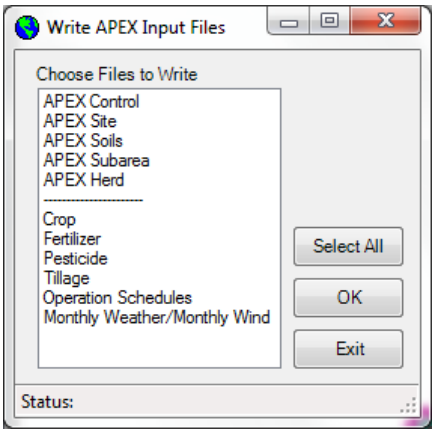

Figure 5.20 Write APEX Inputs Files selection window

SECTION 6 Make APEX Model Run

Now that all input files have been written by ArcAPEX, select **Make APEX Model Run** under the **Run APEX Model** dropdown. If all the input files were edited correctly, the default run setup in the Run APEX Model window will be the baseline model (Figure 6.1). Click **Run APEX** to complete the initial model run. Outputs are printed to text files.

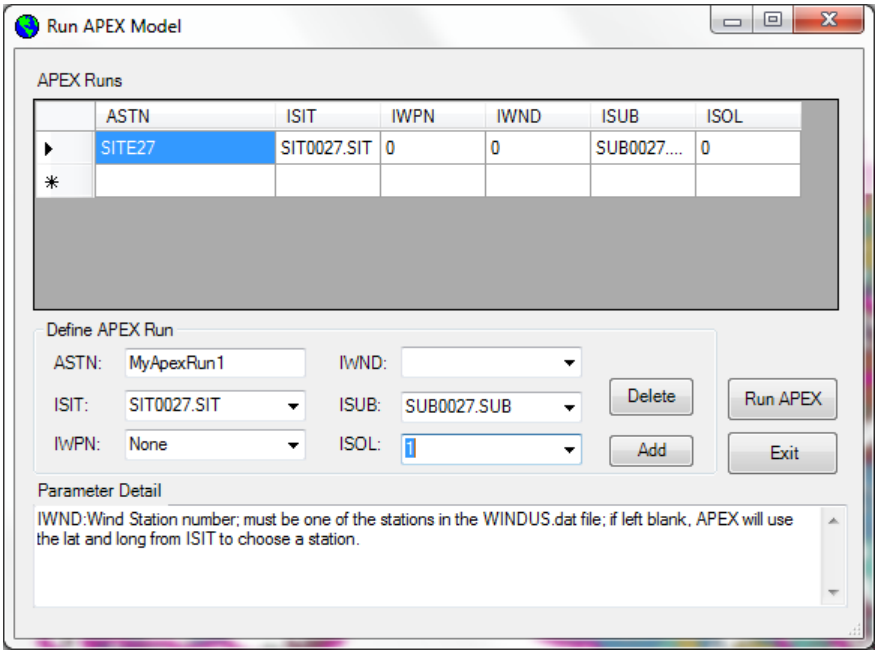

Figure 6.1 Run APEX Model setup window

Note: Once the initial APEX model run has been completed, alternative runs can be made from the Run APEX Model window. Edit any parameters that need to be changed for the new run(s), then set up the new run in the Define APEX run section and click the Add button. Highlight the new run and click Run APEX. Outputs are again printed to text files and will overwrite previous output files.

Appendix A User defined weather data

When using user-defined weather data for an ArcAPEX project, the data must be formatted as listed in the APEX0806 User Guide (2.11, p81-82). The easiest and most accessible way to accomplish this is to input all the data you have collected into a spreadsheet program like Microsoft Excel, leaving empty columns for any data you do not have.

For the example data, **weather1.txt**, we have maximum and minimum temperature (TMAX and TMIN) and precipitation (PRCP). Looking at the user guide, you see that between the day and TMAX is solar radiation (SRAD). Because of this, after importing and converting the data from text to columns in Excel, you must insure that there is an empty column there. Once you have the available data in Excel with any empty columns between existing data, it is now ready to be formatted.

In order to achieve the format needed using Excel, you must make sure the dates (year, month, day) and the data are right-aligned. Next set the column pixel widths in Excel to match the number of columns given in the user guide for each variable by right-clicking on the column and setting the "column width." For year, you can include cols. 1-2 (the 2 blank spaces) by setting the pixel width to 6. Month (cols. 7-10) and Day (cols. 11-14) are each 4 text columns wide, so their column widths should be set to 4. All of the other data columns, as well as any empty columns you have left for spacers, are 6 text columns wide and therefore should be 6 wide in Excel.

Once you have finished formatting column widths, the file must be saved as "Formatted Text (Space delimited) (***.prn**)" in Excel. Excel will alert you that saving in this format may cause some features to be lost and ask if you want to keep using that format. Click yes and then close the file without saving.

Now, change the extension to **.dly**.

If you have multiple weather data sources, repeat these steps for each station to be used.

Next you need to create a pointer (text) file to be used in ArcAPEX to tell APEX where your data file is and what it is called. This file must be in the same folder as your data file. The example, Table A-2, shows the basic, comma delimited file structure.

ID is a generic ID #, this should be unique for each data file used and numbered from 1 to n. This is important as it is how APEX keeps up with which weather data file is being called. Next is your file name including extension (***.dly**), followed by latitude (LAT) and longitude (LON) of the stations in decimal degrees.

Once all stations are in the pointer file (location table), it is input into ArcAPEX.

Table A-2 Weather station pointer file example

ID,FILENAME,LAT,LON
1,weather.dly,44.128,-123.221

Appendix B Lookup Tables

Look up tables should be saved as a comma-delimited (CSV) file in the format below.

For crops, refer to your map data tables to match the numeric value to the crop. You will have to begin the manual matching process as described above to get a list of the crop codes. It is recommended you create a lookup table in case you need to restart at any point during the Land Use/Soils/Slope Definition step.

For soils, the STATSGO S5ID values are available in the Access database located in the C:\APEX\ArcAPEX\Databases folder, entitled SWAT_US_Soils. Simply match the soil types from the SSURGO data table from your map to the S5ID in the database. It is easiest to begin with matching the first few letters of the MUSYM (map unit symbol) to the SNAM (state name), using the NRCS soil surveys as a guide to soil names in the state in which you are working. More information can be obtained at the NRCS website at http://www.fws.gov/stand/standards/soiltool.html.

"Value","S5ID"
1,OH0262
2,OH0262
3,OH0047
4,OH0262
5,OH0047
6,OH0047
7,OH0047
8,OH0257
9,OH0257
10,OH0257
11,OH0257
12,OH0257
13,OH0257
14,OH0048
15,OH0048
16,IL0040
17,MI0042
18,MI0042
19,DC0026
20,MI0042
21,IN0014
22,OH0060
23,OH0060
24,DC0038

Appendix C To append a table of customized user soil records to the usersoil table

The ArcAPEX Parameters Database (**APEX.mdb**) installed with ArcAPEX contains a user soils database table (*usersoil*) with required fields. Users may import an entire table of customized user soil records (such as the one downloaded with SSURGO maps or manually measured) using the ArcToolbox Append tool, or manually add records directly to the usersoil table in the APEX.mdb database.

To use the **ArcToolbox Append Tool**:

1. Select the **ArcToolbox Append** tool (Figure C.1)

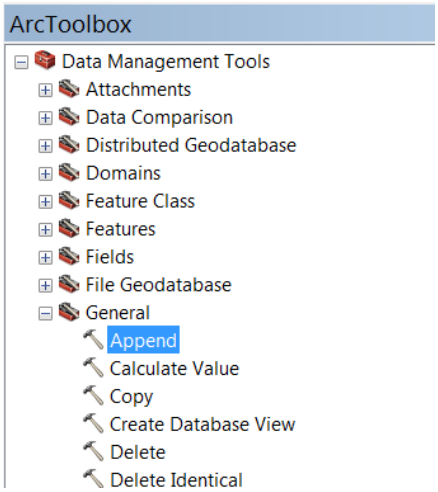

Figure C.1 ArcToolbox

2. Launch the tool and select the customized table of user soil records you wish to append as the Input Features and select the ***C:\APEX\ArcAPEX\Databases /*APEX.mdb/usersoil** table as the **Output Features** (Figure C.2). Click **OK**.

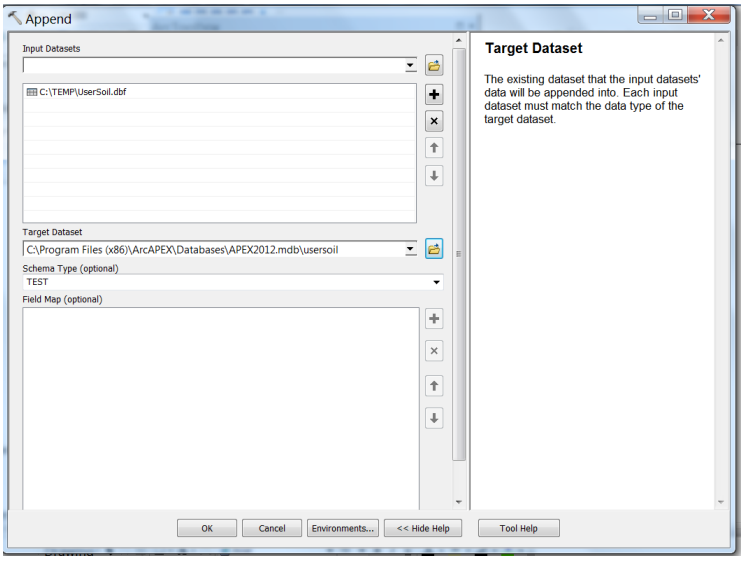

Figure C.2 The Append dialog box

The new records will be added to the *usersoil* table in **APEX.mdb** (Figure C.3).

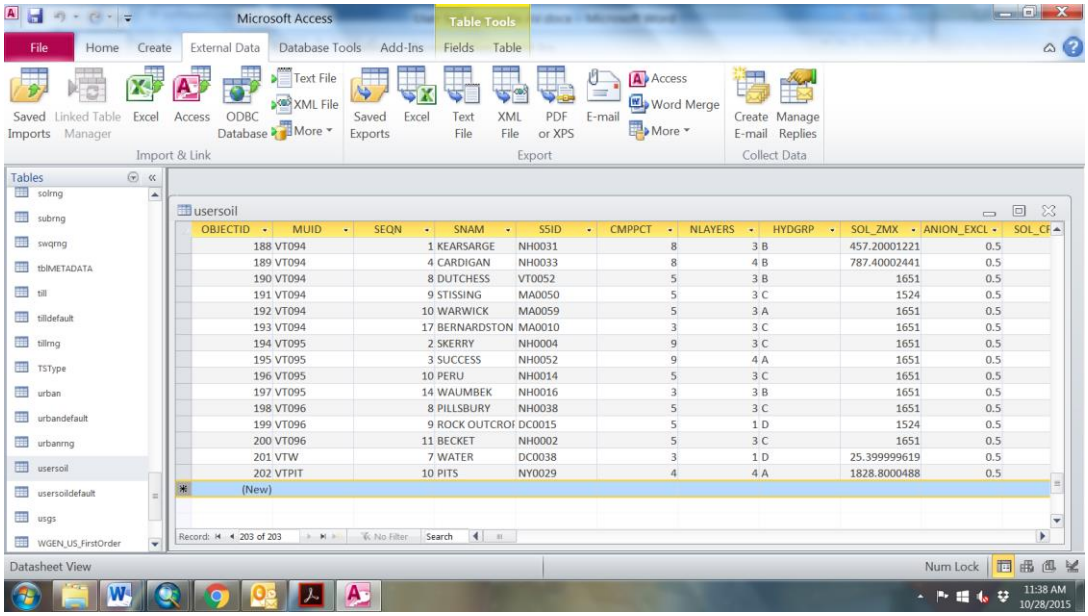

Figure C.3 The usersoil *table in APEX Access database*

Note: The table of new user soil records must conform to the same field structure as the usersoil table in the **APEX.mdb** database. If the table structure is not the same, then the new records may not append correctly and may result in errors in the ArcToolbox append operation.

To append customized soil records to the usersoil table, several options are available. NOTE: Microsoft Access must be installed on the local computer

1. Open **APEX.mdb** and open the usersoil table (Figure C.3)
2. If the data are already in Excel format, copy the records and highlight the entire *new* record row of the usersoil table and paste (Figure C.3). However, soil the Excel records must conform to the same field structure as the usersoil table in the **APEX2.mdb** database.

   OR

3. The usersoil table may be exported as an Excel file using the External Data Export Excel option, appended in Excel and then imported using the External Data Import Excel option (Figure C.3).

Note: The imported table cannot be saved with the same name as an existing table, so the original user table should be renamed and kept as a copy.